



VULNERABILITY ASSESSMENT OF RC FRAME STRUCTURES USING LIGHTWEIGHT SANDWICH FLOOR-SYSTEMS

Michael MISTLER¹, Juan Puig VALLS

SUMMARY

This paper describes a research project whose objective is the investigation of the applicability, implantation and the vulnerability assessment of reinforced concrete (RC) structures using lightweight sandwich floor-systems.

The sandwich panels consist of two thin but strong faceplates called skins, which are supported by a thick but relatively weak core. The skins are composite laminates with a $[0/90]$ lay-up, using E-Glass fibers and resin-epoxy matrix. As this high performance skin composite laminate is fairly expensive, a cheaper and lightweight core material is exploited. The main advantage of this sandwich construction from the structural point of view, is a mass reduction of more than 70% in comparison to traditional European floor-systems. As a consequence, the design requirements in terms of both dead weights and seismic forces are significantly reduced. Furthermore, there are several advantages associated to rescue operations in case of earthquake: facilitating rescue operations since the use of lighter equipment would permit a short response time, reduction of the number of casualties associated to floor-system collapse and its removal as well as reduction of the amount of generated debris. The global ductility of the frame structure is deemed to be uncertain due to the brittle behaviour and high strength of the sandwich slab. This paper intends to evaluate the structural behaviour and the influence of the sandwich slab on the entire structure by considering a full-scale test structure as a sample. Numerical simulations are carried out using the Finite Element program ANSYS, considering material and geometric nonlinearities. The different behaviour of frame structures without and in combination with sandwich floor-systems is investigated and the collapse scenario is anticipated. The damage propagation is investigated exemplarily on the micro level (local joint damage/buckling) as well as on the macro level (entire structure). Finally a procedure to establish a drift-safety relationship in order to determine the slab vulnerability has been developed.

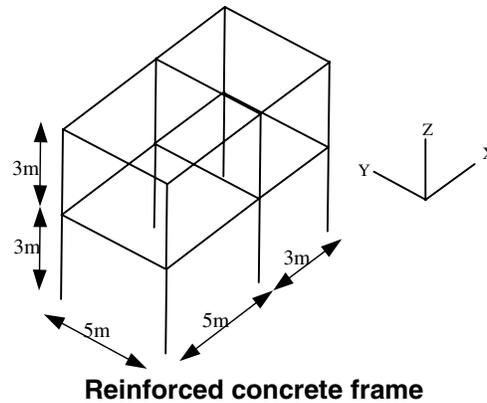
INTRODUCTION

The present study is based on the analysis of a specific sample structure. The main advantage of exercising this special sample structure for this study is that it is going to be built and tested at real scale, thus permitting the eventual validation and calibration of the FE model developed in this study once the test is conducted.

RC frame description

The full-scale test structure consists of a RC frame where sandwich panels are used to configure the floor-system. The geometry of the RC frame consists of a two-storey reinforced concrete frame with two bays of five and three meters in the longitudinal direction and one five meter bay in the transversal direction. The storey heights are three meters.

¹ Aachen University, Chair of Structural Statics and Dynamics, Germany



Columns and beams are designed according to Eurocode 8 [1] considering the RC frame as the only lateral load bearing system. The cross-section of the longitudinal beams in the x-direction is 20 cm in depth and 30 cm in width with 3 bars of 16 mm at top and 2 at bottom. The transversal beams (in the y-direction) have an additional bar of 16 mm at top. The columns have circular cross-sections with 8 bars of 12mm. Additional reinforcement is provided at the joints, splices and anchorages. The slab vulnerability is studied for one lateral loading pattern, only in one direction and with a unique pattern distribution. The load is applied in the longitudinal direction (x-direction) at the joints with the lateral load at the top storey twice the load at the first storey. The boundary conditions of the RC frame are fixed column bases.

Sandwich slab description

Composite materials are characterized by low strength-weight ratios. In fact similar mechanic properties to that obtained with steel can be achieved using composite materials with weight reductions of 5 to 7 times. They are really two materials combined in one: very strong fibers that reinforce weaker matrix materials, which not only bond together the fibers, but also protect them. The volume of fibers with respect to the matrix or fiber volume ratio determines the strength of the composite material.

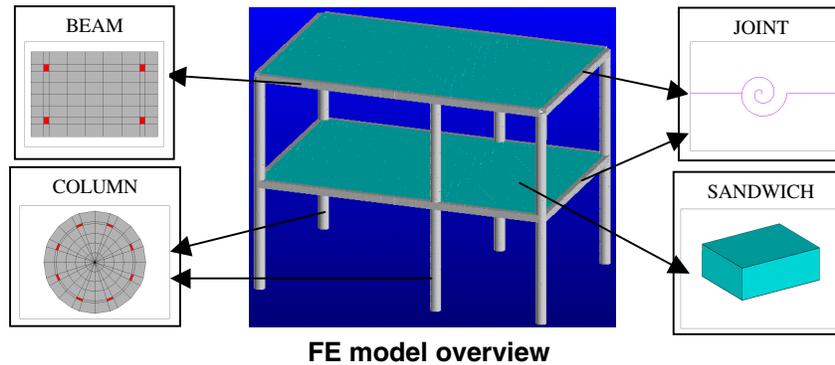
These composite materials are manufactured in laminas, that means, thin plies in which the fibers run in one unique direction. The lamina is very stiff and strong in the fiber direction, but in the transverse direction to the fibers the lamina is weaker, as the properties depend mainly on the matrix. For this reason laminas, each with different fiber orientation, are stacked to form laminates. The fiber orientation of each lamina component of the laminate defines the laminate “lay-up” or stacking sequence. For example a laminate with a [0/90] lay-up is made by stacking a [0] lamina and a [90] lamina, where 0 and 90 indicate the fiber orientation.

Composite laminates have been used in the recent years in the construction industry only for rehabilitation. This project proposes the use of composite materials in floor-systems using a sandwich construction. The sandwich construction consists simply in the use of two outer strong laminates, called skins, which are separated by a weak and cheap core material. The sandwich construction has a larger bending stiffness than the skin laminates simply bonded together, since the moment arm is greater. Conversely the sandwich panel stability and shear capacity depends on the core material. Here, the exercised sandwich panel is 20 cm thick, with 2.18 mm thick skin laminates each.

The experimental data available is basically the result of testing the composite as a laminate with [0/90] lay-up. Testing revealed a linear behaviour of the composite laminate until failure. The stress-strain relationship if given a linear trend achieved values of the correlation factor R^2 of 0.99, either in tension or compression. The behaviour under shear is not so linear, with a value of R^2 about 0.92. Nevertheless, the shear stress-strain relationship has been assumed to be linear.

FULL SCALE STRUCTURE - FEM MODEL DESCRIPTION

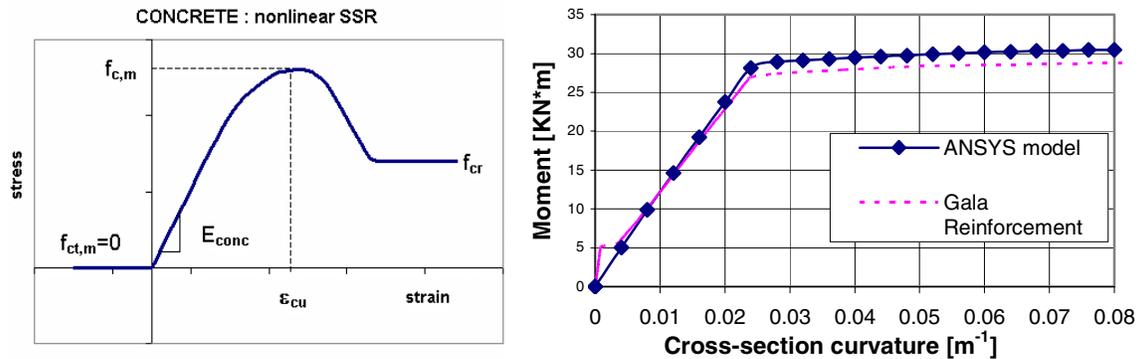
The FE model of the full-scale structure consists of elements representing the reinforced concrete members, the sandwich floor system and potentially the joint between these two.



Reinforced concrete members

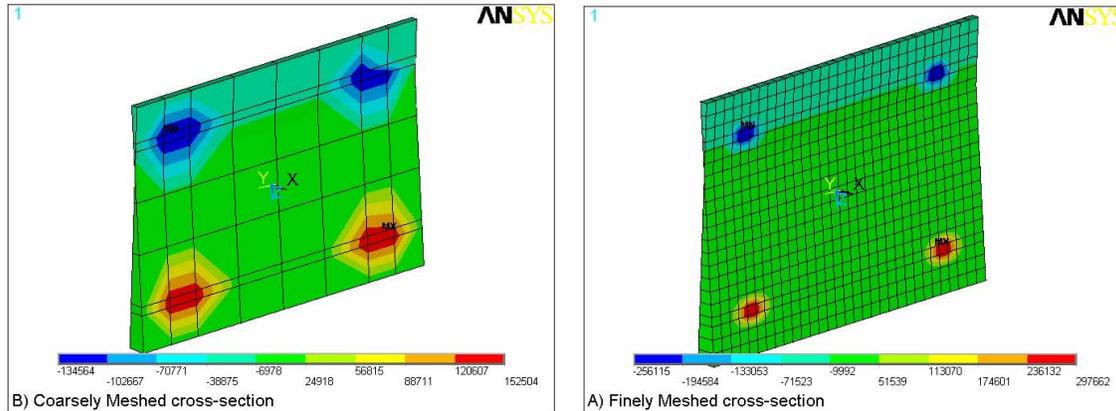
For the reinforced concrete members it is important to be able to simulate the non-linear behaviour including plastic hinges generation. For this purpose, a fibered beam with non-linear concrete and steel materials is employed. The fibered beam consists of a section made out of an array of bonded fibers so that for each of them a different material model can be assigned. Therefore a perfect bonding is assumed between fibers. For each finite element a certain cross-section with fibers of different materials can be defined. The resultant forces of a cross-section are the result of the contribution of each fiber force. The latter is simply obtained from plugging the strain in the constitutive relationship of the considered fiber. The advantage of using a 3-D beam element is not only lower computational time, but also much easier modeling and review of load input or results compared to that of a model built out of 3-D solid elements. Variables such as moment, curvature, axial force etc. are explicitly in models. The disadvantage is that the availability of stresses at a certain cross-section is limited, as opposed to a model using 3-D solid elements.

The structure under scope is to be tested until failure, hence material nonlinearities are not negligible if reliable results are to be achieved and if the generation of plastic hinges upon loading of beam members are to be simulated. Here, a user-defined material model is used that accounts for all possible concrete material singularities as shown in the following figure (right hand side). The steel material model is simply an isotropic perfectly plastic model. To verify the accuracy of the reinforced concrete frame response the sectional behaviour is checked by the moment-curvature obtained by this model compared with that computed with a commercial software. The comparison between both shows a very satisfactory agreement (left hand side).



Stress-strain relationship of user-material model and moment-curvature relationship

The stress distribution of the cross-section is erroneous: Concrete cells surrounding rebars bear excessive stresses and conversely steel cells representing rebars show moderate stresses. Finer meshed cross-sections present higher stress concentration only in the rebar cells or fibers, thus stresses in steel and concrete cells are more realistic. Nevertheless, finer cross-sectional meshes do not provide any additional accuracy to the moment-curvature sectional behaviour. Thus there is no need to use them in an analysis in which stresses are not of interest, but the overall frame response. The next figures show the comparison of two differently meshed sections where positive stresses indicate tension.



Axial stress distribution of: a) Finely meshed section; b) Coarsely meshed section

Sandwich floor-system model

To model the slab 4-noded nonlinear layered shell elements with sandwich construction capability are used. As for the mesh refinement of the slab, it depends on the refinement of the RC frame. Stress concentrations take place at the corners next to the column. For this reason it is appropriate to provide finer refinement for these locations, where the gradient of the stresses is higher.

The materials of the sandwich slab are linear. For this reason it is important to check if the slab elements have actually failed and therefore failure of these elements need to be considered somehow. Once the materials conforming the sandwich slab fail, the shell elements do not transmit or bear loading any longer. To overcome the problem of the lack of failure criteria based on element table output of the used FE program, a user-programmed macro is used to generate such element table. It consists basically on the finding of the maximum ratio of actual stress or strain to the allowable stress or strain. Thus when the failure criterion output ratio is greater than one indicates material failure. This process is carried out for each layer and for some locations within the layer; for instance, at the top layer only stresses at the top are checked. Since the materials are assumed to be linear, it is already known which of the ratios is going to dominate: stress ratio for the laminates and strain ratio for the core.

A alternative way to obtain the failure criteria for each element is taking advantage of the ANSYS built-in check by means of the "FC" (Failure Criteria) command. For a specific material under certain loading it is possible to know if those ultimate stresses or strains are being exceeded. However, the value provided is only a nodal value that depends on the layer and its location within the layer: Different values are obtained at layer 1, 2 or 3 as well as if they are evaluated at the top, middle or bottom of the layer. That means that element tables are not possible to be defined. Also, the maximum value of the failure criteria throughout the shell element is not provided. Besides, this FC-value can only be retrieved using plotting commands or by single nodal data.

A second macro using the FC command can be written to check the previous one. This macro is much slower than the first, given that nodal solution data is used and needs to get averaged. It also

requires that single elements are selected when the command “GET” is issued, otherwise averaged results are retrieved, given that each node is shared by several elements (not conservative).

The macro creates blank element tables that are filled in and the maximum is found from the failure criteria values for each layer, for each location within a layer and for each failure criteria (maximum stress or strain). The error found between these two procedures is less than 10%. For more details and input files of both macros refer to [2].

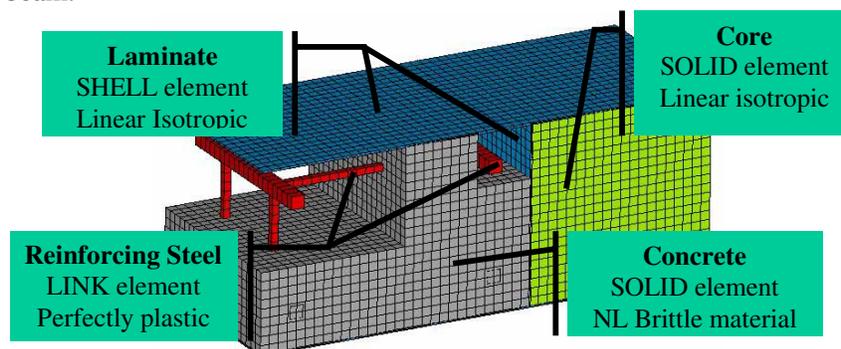
Joint model

The joint between the sandwich slab and the concrete beam is a very complex structural member. The response of the joint as well as the failure modes are unknown. The intention here is to establish a macro model that represents the joint. Such model is then to be implemented in the full-scale structure. In order to develop this macro model, an FE simulation is performed by means of a micro analysis of a single joint. The joint has been modeled assuming the following: Only those failure modes relevant to the present study have been examined, given that the loading has been previously established. Such failure is then studied by the appropriate finite element model. Furthermore, the definitive model has to be implemented in a model where the frame is represented by 2-noded beam elements and the slab by shell elements.

MICRO ANALYSIS OF THE JOINT

FE model description

The micro joint model is 3-D and not 2-D because the transverse reinforcing steel makes the model not symmetric along the beam spanning direction. An initial version of the model is shown in the next figure where the different colours correspond to different materials; taking advantage of symmetry along the beam direction every 10 cm (hoop spacing) and modeling only part of the slab that frames in the beam.



Overview of the joint model

The 3-D parts of the model have to be necessarily modeled with solid elements including crushing and cracking feature. The steel has been modeled with truss uniaxial tension-compression element with three degrees of freedom (DOF) at each node including plasticity and eventually large deflection capabilities. In order to model the laminate shell elements are more suitable than solid elements. A 4-noded shell has been chosen for this model including full nonlinear, suitable for large strain and deflection applications, layered section definition as well as failure criteria. The model is a shell that represents the [0/90] laminate as a whole, representing the smeared material properties. Such a model is assumed to be linear isotropic. The core is defined as an isotropic linear elastic material. As failure criteria the maximum strain failure criterion and the maximum stress failure criterion have been employed.

Joint FE analysis

The FE model of the joint is considered to simulate the response under certain loading as well as to predict determined modes of failure. Finally, a model that represents the joint should be established.

The joint is studied assuming certain failure modes as a component of the entire structure; such failure is then studied by the appropriate finite element model. The behaviour provided by such model is then summarized in terms of beam and shell elements degrees of freedom so that it can be implemented in the full-scale structure. The implementation of the joint model in the full-scale structure is determined to be done by means of a nonlinear spring element. This way the author takes advantage of ANSYS tools especially concerning post-processing; for instance, propagation of failure can be easily monitored using element status-variables.

Identification of Joint Failure Modes

The identification of the expected joint failure modes will serve as starting point for the joint model conception. Hence it will determine geometry, suitable loading and boundary conditions, symmetry considerations etc. Notice that the scope of this micro-analysis extends only to those local effects exclusively within the joint and those structural elements next to it. Thus, buckling phenomenon is of interest only at the joint, and the structural buckling of the slab can be assessed from the full-scale model analysis.

Taking into account that the structure is assumed to be loaded only in the longitudinal direction, it is clear that there are two different types of joint failure depending on which of the two sides of the slab is being considered. The joints that are located along the short beams (transversal side) will mainly transmit moment from the slab to the beam, given that the beam is framing into the column and the column imposes a rotation to these beams. It is due to the locations where plastic-hinges have been developed: At the base of the column and in the longer adjacent beams. Therefore the column describes a solid rigid rotation about its base as the drift of the frame increases, and so does the rotation of the short beams framing in the column. As for the joints along longitudinal sides the failure mode is less intuitive than the previous. Basically torsion from the slab is transferred as bending to the adjacent longitudinal beam, in other words, the slab helps the beam to undergo bending by transmitting torsion moments to the slab. This previous explanation could lead to think of the suitability of using torsional springs to model the joint response in transmitting torsion from the slab in terms of bending of the beam. However, if very fine mesh of beam elements were to be used, then the beam rotations would no longer be significant and so the use of torsional spring elements would not be a good representation of the joint behaviour. The failure is likely to be the point when the bending is no longer coupled between slab and beam. If the geometry and materials of the joint are taken into account, three failure modes are likely to happen:

- The breakage of the concrete-laminate bond at top and bottom in a shear-friction mode, given that the moments are carried in sandwich slabs by the skins.
- Buckling of the horizontal compressed laminate bonded to the concrete beam as the beam bending progresses.
- Failure of the vertical bonding strip located between the core and the concrete.

Experimental data would be at this point extremely valuable to determine the most likely failure mode; due to the lack of testing, the failure mode is not only non-intuitive but also undetermined.

The problem of modeling this joint is simply that this failure mode cannot be represented by a single DOF element. Furthermore, if the full-scale model is built by shell and beam elements the joint behaviour cannot be modeled given the lack of sufficient DOFs. For this reason it has been decided not to model the failure of the joints along the longitudinal sides. Only by taking a look at the computational time inverted on the calculation of the developed joint models gives an idea of the computational cost, if the full-scale structure were to be modeled entirely by solid elements. Consequently, joints along the longitudinal beams are not modeled. From now on the rest of the study refers exclusively to the joints along the short beams.

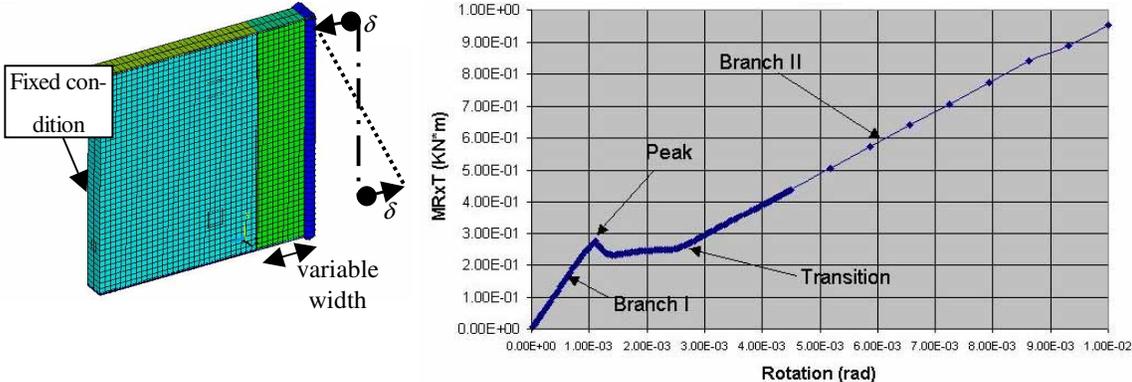
Modeling of joint bending transfer behaviour

The loading of the model is such that represents the previously discussed failure mode. Since the objective is to develop a moment-rotation relationship to be implemented in a nonlinear spring ele-

ment, the loading consists of an imposed displacement at the top and bottom skins of the slab. From the reaction forces, obtained at the points where the displacement is imposed, the reaction moment can be calculated. The loaded slab section is forced to remain plane. Appropriate boundary conditions are used to model both interior and the exterior joint: The boundary conditions of the model are, at the cut-end of the half-beam, a fixed condition and at the front and back of the model constraining in order to account for the symmetry along the beam spanning direction.

There are four types of contacts in the model: concrete-laminate, concrete-steel, laminate-core material and joint-laminate and slab-laminate. The last contact, laminate-laminate, since it is performed by adhesive bonding, it has been assumed that there is a perfect bonding throughout the analysis; meaning that, in case of failure, the laminate itself would fail instead of the bond between laminates. For concrete-steel contact no-slippage has been assumed, since this effect has little interest in this study. As for the other two, no data is available about the behaviour of this bondage. However, from testing observation can be concluded that the bondage is probably stronger than the materials taking part in the bond. Therefore for the concrete-laminate and core-laminate contacts it has been assumed also perfect bonding: The materials at the contact are subject to failure, but not the bondage itself. Perfect bonding or no-slippage condition is modeled by merging coincident the nodes from the elements throughout a contact. Also by defining appropriate brittle material models (in the case of concrete) or by checking allowable stresses and strains (failure criteria of laminates and core materials) the failure at the interface is considered, although it depends on the adjacent materials.

Upon some testing, following facts have determined some changes in the initial lay-out of the described joint: There is no steel interaction with the model. This can be observed by looking at the deformed shape, which does not affect the steel elements. Therefore, if a thinner model is used, the same results per unit length are obtained saving a great deal of computing time. Therefore, a 2 cm thick model has been definitively used. Furthermore, when the skin laminate is about to fail, the core is not failing and the concrete has already crushed-cracked at its most loaded edges. This state determined the maximum displacement level to be applied. Further loading would imply slab failure and thus it is of no interest. Finally, an important assumption is that the slender laminate parts of the joint are not subject to buckle because only a linear geometric analysis can be performed with the actual material model; i.e., the concrete brittle material leads to erroneous results if used in conjunction with the large-deflection solution option. Thus, it is necessary in order to predict buckling behaviour.



Joint model for bending transfer behaviour and typical moment-rotation relationship

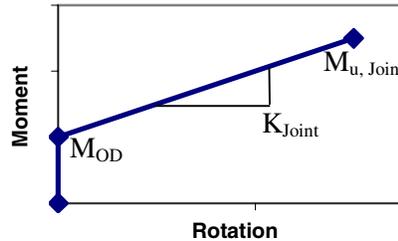
For this reason a different model has been used to model the eventual buckling of the slender parts of the joint. Taking into account these considerations, the model shown in the next figure is the definitive model developed to study the bending joint behaviour. Note that the slab width considered is in any case smaller than the part of slab where the additional bonding laminate strips is covering.

The resulting moment-rotation relationship for a certain width of the slab, shows typically the plot output of the figure on the right hand side. During the first steeper slope of the plot (branch I) the imposed δ displacements produce a deformation in the slab, while the beam remains undeformed. The peak that follows this first branch corresponds to the concrete local failure at the loaded edges of the beam. Up to this point the imposed displacement deforms the beam edges while the slab remains now undeformed. Upon variation of the slab width, the moment-rotation relationship shows different changes:

- The first branch slope depends strongly on the slab width, given that the slab is the part of the joint undergoing deformation. At different widths, the rotation is different since the length along which the curvatures of the slab sections are integrated is longer.
- The moment level that determines the onset joint damage, that is, the peak marking concrete crushing/cracking is approximately constant for different widths.
- The second branch, in which the slab no longer participates on the deformation, has a slope quite constant. It is slightly stiffer when the width of the slab decreases; because the axial deformation of slab skins is then smaller.

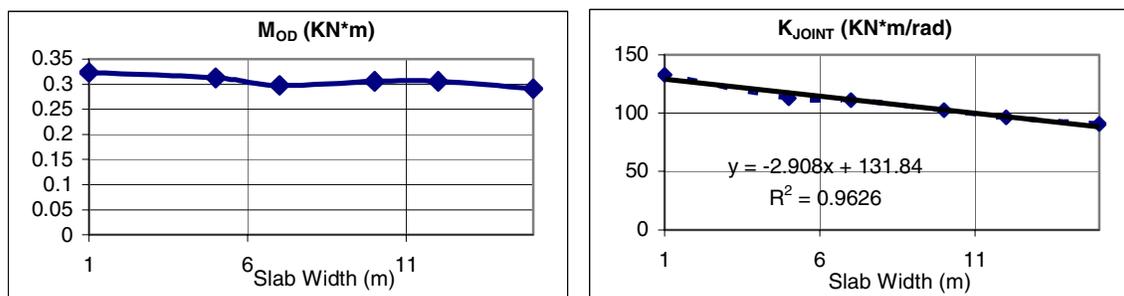
Micro joint model

At the sight of the previous considerations, the model of the joint to be implemented in a nonlinear spring element with a single degree of freedom is shown in next figure.



Joint Model for a nonlinear SDOF spring

The model parameters are the moment of joint onset damage M_{od} , the stiffness of the damaged joint K_{joint} and the moment of joint failure $M_{u,joint}$. Up to a moment level of M_{od} the joint behaves as a rigid level, as if slab and beam rotation were coupled. The rigid behaviour is theoretically represented by an infinite slope or vertical line on the moment-rotation relationship. Once M_{od} is reached, the joint becomes semi-rigid. In order to make the parameters independent from the FE model geometry, i.e. the slab width, an analysis of the joint with different slab widths have been executed.



M_{od} and K_{joint} adjustment with respect to slab width variation

The mean value of M_{od} , times a factor of 50 to take into account a joint of one meter long, is 15.3 KN·m. Similarly for K_{joint} , but taking the value corresponding to a width of slab equal to zero, yields 6592 KN·m/rad.

As for the ultimate moment of the joint $M_{u,joint}$, the analysis is stopped when the displacement imposed produces the skin of the sandwich slab to fail. Thus, if K_{joint} is not considering the piece of

slab strengthened by the bonding strip laminates, and this laminate fails before the joint does, then this type of failure is already taken into account on the full-scale model. Notice that the full-scale model includes the strengthened strips next to the joint, since they are not included in the joint model. On the other hand, buckling effects are not included in this model. A failure mode of the joint by buckling requires a different model which allows the large-deflection solution option being used.

Modelling of joint buckling

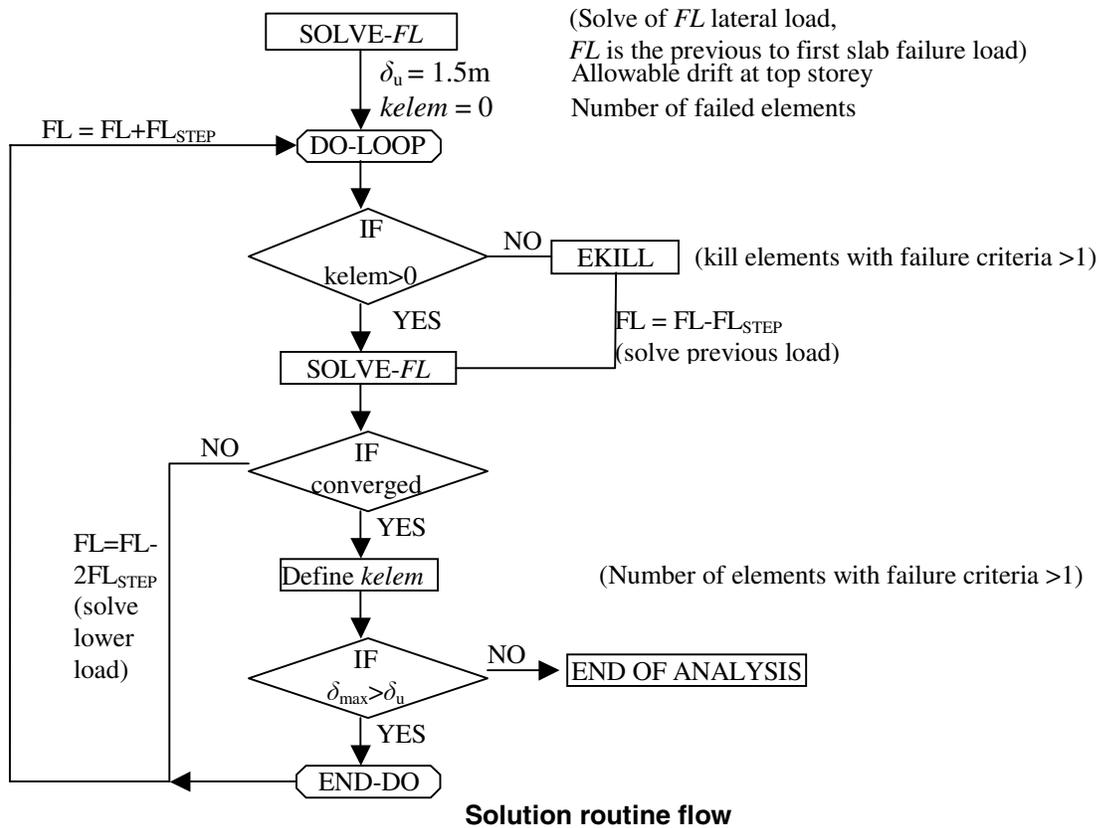
A specific model is also used in order to study buckling of the slender parts of the joint, given the incompatibility of ANSYS concrete supported material model and the solution mode that takes into account large-deflection effects. Once the onset damage has occurred, i.e. cracking/crushing of the concrete beam edges, then the expected ultimate failure of the joint is the buckling of the laminate in compression: This laminate is transmitting in combination with the bottom laminate, the entire moment transmitted by the slab. Furthermore, the laminate buckling is restrained by the concrete bond-age, which in tension is specially weak. This phenomena is investigated based on a model slightly modified. But since the bending moment that the sandwich slab can carry (including buckling) is less than the predicted buckling moment there is no need to implement a moment of failure in the joint model.

FULL-SCALE STRUCTURE ANALYSIS

The main objectives of the FE analysis are the establishment of the influence of adding a sandwich floor-system to the RC frame as well as the assessment of the slab behaviour in terms of safety reduction due to joint and slab damage. For this task a pushover curve of both systems (with and without slab) is performed and for different stages of lateral force level, the slab and joint damage is evaluated.

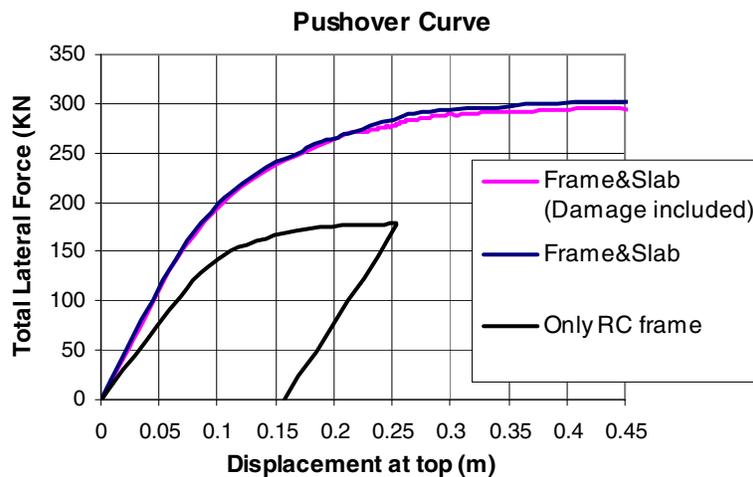
The analysis of the test structure is defined to be a force-controlled pushover test. The forces are applied to the first and second storey joints in one side, with the forces at the top storey twice as those acting at the first storey. The analysis is divided into two main parts: One in which only the reinforced concrete frame is studied and a second in which the sandwich floor-system is included. Both analyses are executed in order to compare the influence of the addition of the sandwich panel floor-system.

In the present study it has been assumed that when the materials conforming the sandwich slab fail, the shell elements do not transmit or bear loading any longer. No material model, other than user-defined, can represent such behaviour. However, the majority of elements in ANSYS support “Birth and Death” feature which permits the user to reduce the material properties of specified elements by a certain factor. The solution routine has special interest because it represents a general procedure on how to use “Birth and Death” feature with analysis-restarts on those elements exceeding some element-table-based values.



An inherent problem of this analysis is the convergence problems caused by the elements which include nonlinear (changing-status) formulations, like the bilinear joint elements and using “Birth and Death” feature which introduces not only a change in status of the element, but also discontinuities in the FE model. Furthermore, there are some conflicts between processors. For more information refer to [2].

Force-controlled pushover

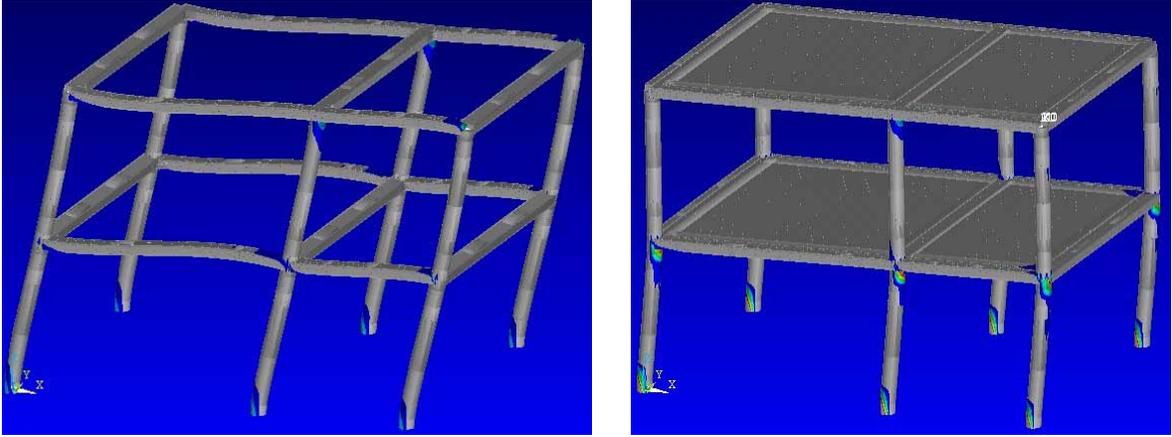


Comparisons of pushover curves: with and without sandwich floor-system

The pushover curve is obtained by plotting the summation of the reaction forces in the x-direction at the base of the column (total reaction forces) versus the drift at the top storey. The comparison of obtained pushovers of the structure with and without sandwich slab is given in the shown figure.

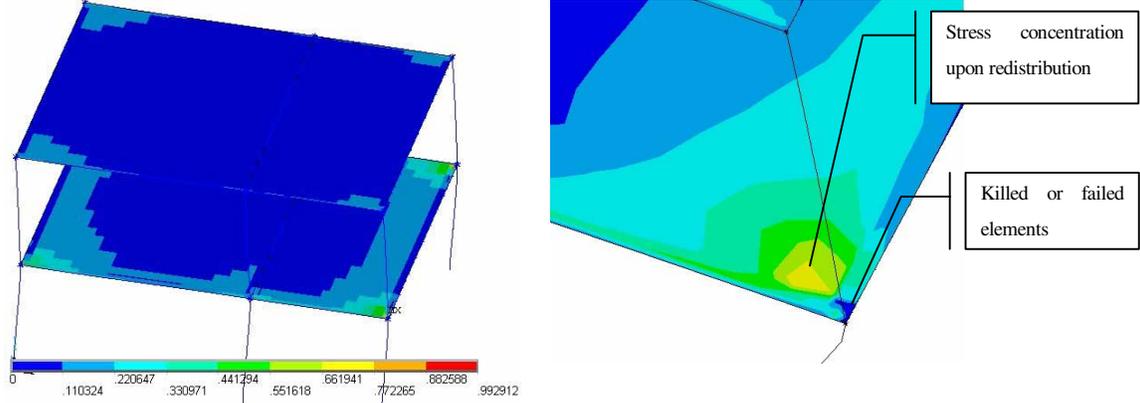
In the system disregarding the slabs plastic hinges are developed at the base of the columns as expected. Plastic hinges of the first storey develop at beams and not at columns, therefore the EC8 provision about weak story failure mechanism is satisfied [1]. This can be also assessed from the solid rigid rotation of the columns about their bases, which remain straight.

In the new structure configuration including the floor-system, the plastic hinges are located at the columns and not at the beams. The analysis shows that the slab forces the plastic hinges to develop at the column. On one side, the transverse beams torsion is restrained by the slab, and as the beam is framing into the column, the latter has to undergo larger moment levels to remain straight. On the other side, the longitudinal beams are stronger, as their bending resisting capacity is enhanced by their rigid connection to the slab; in other words, the slab is “helping” the beam.



Plastic Hinges of the RC frame in combination without and with a sandwich floor-system

Failure of the slab occurs locally next to the columns. Nevertheless, very few elements are failed and the failure is of little extension. The following figure shows the failure criteria. Values greater than 1.0 mean that the element is exceeding the allowable predefined stress or strains, while zero-values mean that the element stresses and strains are zero.

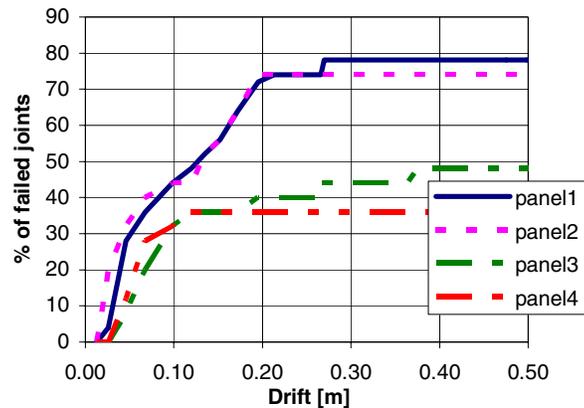


Failure Criteria distribution at $\delta=0.47m$ and a detail of corner showing failed elements and the failure criteria redistribution upon element killing

On the right side a detail of the failure criteria (FC) is shown at a stage in which the horizontal branch of the pushover has been reached. The failed elements can be identified by FC values equal to zero. It is notable how, once one element is killed, the elements next to it pick up the stresses pre-

viously borne by the failed element; thus resulting in an increase of the FC value in those elements next to the failed one. This stress redistribution determined the failure propagation throughout the slab.

Finally, the damaged joints of each slab can be monitored. Most of the joints get damaged at a low lateral force, and the joint damage progresses from the columns to the transversal beams midspan. At the horizontal branch of the pushover the majority of the joints are damaged, i.e. they no longer behave as a rigid joint, but as a semi-rigid joint. The next figure shows the progression of joint damage along the transversal sides as the lateral force increases.



Percentage of damaged joints along the transversal sides of each slab panel versus drift

VULNERABILITY ASSESSMENT

In order to establish the vulnerability of floor systems made of composite materials using a sandwich construction the following procedure is presented:

- establish the relationship between slab safety reduction and roof displacement,
- determine maximum roof displacement due to a certain earthquake loading,
- associate slab safety reduction to annual probability of exceedence by obtaining maximum roof displacements for earthquake loadings with different return periods.

Notice that the relationship between the slab safety and the roof displacement depends indefectibly on the structure geometry, thus not allowing establishing a general relationship for any frame structure configuration, such as the typically used interstorey drift for framed structures vulnerability. However the procedure presented to establish the slab safety reduction, given the slab damage is a general procedure, which could be used with lateral or any other loading pattern.

Slab safety estimation

In order to evaluate the slab damage at different stages of lateral displacement the slab safety reduction is estimated according to certain damage indicators. Assuming a lateral loading, at each point of the pushover curve the slab damage can be evaluated in terms of the failed elements and damaged joints. The slab damage of each point of the pushover can be then included in a second analysis in order to define a damage indicator, which can be evaluated using three different approaches:

1. Serviceability approach: This approach is based on the residual deflection under a uniformly distributed load of reference. The obtained deflections are then compared to the first deflection, considered as a reference of the undamaged floor-system.
2. Ultimate state approach: This approach computes the slab capacity or maximum allowable uniform load borne by the slab at different stages of damage. The slab capacities are compared to that slab capacity corresponding to the undamaged slab floor-system.
3. Modal-analysis based approach: The idea of performing a modal analysis is that the resulting frequencies are function of the stiffness of the system. In the structure scope of study, as the damage

progresses due to increasing lateral loads, the overall stiffness decreases and if the mass remains constant the frequencies also decrease.

If the damage indicator of a certain point of the pushover curve is compared to that of a structure with no slab damage at all, it gives the slab safety reduction undergone by the slab at that point of the pushover curve.

Capacity spectrum method

The capacity spectrum method, described in [3], permits the calculation of the maximum structural displacement for a certain earthquake loading. The method consists of representing both the pushover curve and the response spectrum in response spectral ordinates or ADRS format (acceleration-displacement response spectral ordinates). The intersection of both curves defines the performance point.

The capacity spectrum method in order to estimate the area of such loops, assumes that the pushover curve can be represented bilinearly, thus being the loops simple parallelograms. The process is iterative, as the performance point, intersection of the two curves, determines the bilinear representation from which depends the response spectrum reduction (due to damping).

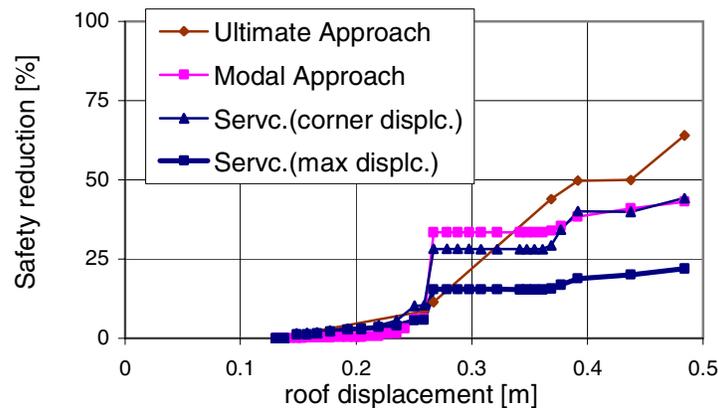
Risk assessment

The risk assessment permits the evaluation of the hazard involved by using sandwich floor-systems in a determined seismic zone. On one hand, the relationship between roof drift and safety reduction for a specific structure can be estimated. On the other hand, the maximum roof displacement can be determined for a given earthquake loading, i.e. for a certain seismic zone, soil condition and return period. Therefore, if different return periods T_R are assumed, maintaining constant the rest of parameters, the safety reduction for such a seismic event can be estimated. For a certain earthquake associated to a T_R , both safety reduction and probability of annual exceedance can be determined. If different return periods are used then the relationship R-Slab Safety can be plotted indicating the risk assumed when using a sandwich slab in a certain seismic zone.

Example of application

With the actual design the failure of the slab is of little extension. This leads to damage indicator ratios between 0.99 and 1.00, i.e. with the actual design the slab does not show an appreciable safety reduction in terms of serviceability, ultimate state and stiffness modal-based analysis. Although the slab is barely damaged, this method is useful to analyse future structures that include other materials or design specifications.

In order to demonstrate what kind of results are obtained from the three different approaches, an analysis considering the maximum allowable stresses and strains to be half the actual has been performed. Therefore, the estimated safety reduction versus drift is shown in the next figure using the three approaches.



Safety reduction estimation versus structure drift

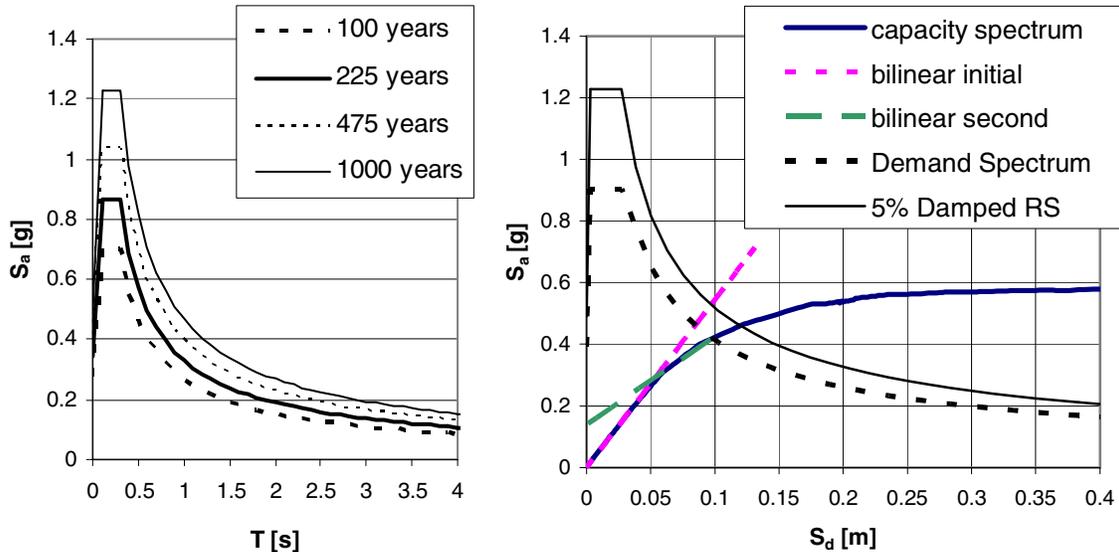
For the serviceability approach, the maximum displacement and the displacement at a quarter of the slab have been used. The latter yields higher safety reductions as it is closer to where damage occurs. It is nota-

ble that the fact that a serviceability-based safety reduction is similar to that obtained from an ultimate state approach can be tricky: it is not qualitatively the same, in terms of safety, a deflection decrease in 50% than a slab capacity decrease in 50%.

The slab undergoes significant drops of safety at drifts about 25cm and 35cm. Note how these results are obtained for the most damaged slab panel, being the damage on the other panels of little magnitude. Thus, a general slab safety rule depending on the roof displacement is not possible.

Capacity spectrum method

In Europe there are several active seismic zones of interest in which the performance of sandwich floor-systems could be investigated. In the present example, the city of Istanbul, Turkey, is chosen. The spectra for different return periods are shown in the following figure on the left side.



Earthquake spectra of Istanbul and the ADRS (performance point finding)

On the right side three curves are represented: the capacity spectrum with its bilinear representation, the 5% damped response spectrum for a return period of 1000 years and the damped-reduced demand spectrum. Notice how the intersection of the initial stiffness of the capacity curve with the 5% damped response spectrum represents an accurate estimation of the performance point (intersection of the capacity and demand spectrum). The performance point defines the maximum displacement of the structure under the specified seismic demand. The results are summarized in table on the right side. Notice that the obtained displacements are quite small, even though the ground motion is significant. The reason is the low weight of the floor-system (about 15 kg/m²) which influences greatly the response of the structure upon ground acceleration. This fact is apparent in the capacity spectrum method when the capacity curve is transformed into spectral response ordinates, since the

Capacity spectrum results

T _R Return Period (years)	δ _{roof} Max. roof drift (cm)
100	6.8
225	8.3
475	10.2
1000	11.9

Probabilities of annual exceedence for various T_R

T _R Return Period (years)	t Time at the end of interval (years)	R probability of annual exceedence (%)	δ _{roof} Maximum roof drift (cm)
100	1	1.00	6.8
225	1	0.44	8.3
475	1	0.21	10.2
1000	1	0.10	11.9

equivalent spectral acceleration is inversely proportional to the dead weight of the building. As a result of the little self-weight of the structure in comparison to the lateral resisting stiffness, the earthquake hazard is not likely to be a critical design parameter for this type of structures.

Risk assessment

As commented in the previous paragraph, this type of structures, at least with the present design and material specifications, does not present a notable seismic vulnerability even for the low roof displacements obtained even for a 1000 years return period. Anyhow, the probabilities of annual exceedance are detailed in the next table.

CONCLUSION

The present numerical simulations shows a powerful tool which permits the prediction of the expected structural behaviour, however there is a limitation in such prediction given that a few assumptions were adopted. For this reason it is recommended to calibrate the model once testing on the subject has been conducted. The most relevant assumptions are the use of fix joints along the longitudinal sides and not considering the compressed skin buckling by delamination in the sandwich panels.

The general behaviour of the structure is greatly determined by the low weight of the structural components. For instance, the dead load is much lower compared to conventional structures; thus representing the live loads is the most important part of the loading. In the case of the seismic loading, the structure shows a very stiff response, result of the fact that the earthquake forces are inversely proportional to the mass of the structure.

For this reason the columns in terms of seismic requirements are actually overdesigned. On the other hand, assuming the model limitations, the addition of the sandwich slab with the present design causes the plastic hinges to be developed not at the beams but at the columns as the structure is loaded laterally. This determines a weak story failure mechanism, which is undesired from the seismic design stand point. So, attending to this, the columns would be underdesigned.

The plastic hinges are developed at the columns because the slab collaborates with the beams monolithically and the slab was intentionally neglected during the RC frame design. If the plastic hinges developed at the beams, it would ensure an appropriate failure mechanism as well as a more flexible system in order to be designed to meet the seismic demand. This can be achieved by: using weaker slab-beam connection, like pinned or even simply supported, reducing the slab capacity and stiffness or using stronger columns or weaker beams.

In particular, the test structure scope of the analysis, with the present allowable stress/strain based failure criterion, shows that damage of little extent occurs locally next to the columns under lateral loading. If reduced allowable stress/strains are used, it is notable that the failure eventually progresses from the parts of the slab next to the column towards the midspan of the transversal and longitudinal beams.

Since the slab undergoes little damage, the slab safety reduction is insignificant. Furthermore, the safety of the slab remains at adequate levels until the damage has grown extensively along the boundaries. Nevertheless, this situation is unlikely for earthquake hazard, given the small maximum roof displacements obtained for this type of lightweight floor-system construction.

ACKNOWLEDGMENTS

This work has been conducted within the Fifth Framework Programme of the European Commission, project acronym "SAFEFLOOR", contract number EVK-CT2000-00020. The financial support provided by the European Commission as well as the co-operation of all project partners is gratefully acknowledged.

REFERENCES

1. Eurocode 8, Part 1.4: "Design Provisions for Earthquake Resistance of Structures – General Rules – Strengthening and Repair of Buildings", 1998.
2. Valls, J.: Vulnerability assessment of reinforced concrete frame structures using composite sandwich panel floor-system, Dipl.-Arbeit, Chair of Structural Statics and Dynamics, Aachen University, 2003.
3. ATC 40: Applied Technology Council, Seismic Evaluation and Retrofit of Concrete Buildings, 1999.